International Journal of Mechanical and Production
Engineering Research and Development (IJMPERD)
ISSN (P): 2249-6890; ISSN (E): 2249-8001
Vol. 7, Issue 5, Oct 2017, 1-12

FINITE ELEMENT ANALYSIS AND MULTIBODY DYNAMICS OF 6-DOF INDUSTRIAL ROBOT

RAHUL ARORA & S. S. DHAMI

Department of Mechanical Engineering, National Institute of Technical Teachers Trainee and Research, Chandigarh, India

ABSTRACT

This paper implements the design structure of industrial robot along with the different transmission components like gear assembly and analysis of complete industrial robot. In this paper, it gives the overview of the most efficient types of modeling and different analysis results that can be obtained for an industrial robot. The investigation is executed in regards to two classifications i.e. the deformation and the stress tests. Solid Works is utilized to design and review the 3D drawing plan while ANSYS Workbench is utilized to execute the FEA on an industrial robot and the designed component. The CAD evaluation was conducted on a disentangled model of an industrial robot. The study includes design and drafting its transmission system. In CAE study static, modal and dynamic analysis are presented. Every one of the outcomes is divided in regard to the impact of the static and dynamic analysis on the situating exactness of the robot. It gives critical data with respect to parts of the industrial robot that are inclined to harm under higher high force application. Therefore, the mechanical structure under different operating conditions can help in optimizing the manipulator geometry and in selecting the right material for the same. The FEA analysis is conducted for four different materials on the same industrial robot and gear assembly.

KEYWORDS: *CAD, CAE, FEA, Robot, Static, Dynamic, Modal and Gear Assembly*

Received: Jul 22, 2017; **Accepted:** Aug 12, 2017; **Published:** Aug 18, 2017; **Paper Id.:** IJMPERDOCT20171

Original Article

INTRODUCTION

An industrial robot is characterized on the bases of specification mainly into payload, reach, precision and repeatability. Depending on their configuration and size, robots can reach a collection of points around them that constitute a workspace [12]. The work envelope of an industrial robot IRB 140 is shown in Figure 1. [8]

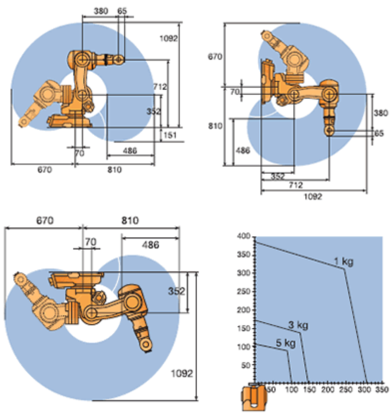


Figure 1: Work Envelope ABB IRB 140 [8]

Robot design depends upon operating conditions such as payload capacity, work volume, accuracy,

maximum speed required etc. These factors influence selection and location of the actuators as well as the structural design of the manipulators, as it has to withstand the stresses and deformations due to dynamics. Therefore, the rigidity of the mechanical structure is very important for accuracy and precision of robot, and hence, the design of mechanical structure of robot is critical.

Robot design may be broadly divided into three parts, i.e. Manipulator Configuration, Forward kinematics and Inverse Kinematics. Robot design consists of selecting structural dimensions, material, motors, sensors etc. The positional accuracy that can be obtained depends upon robot configuration, actuators, mechanical elements and the controller. The mechanical rigidity of the manipulator also contributes towards achieving designed positional accuracy with different payload. Further robots usually perform repetitive tasks and the manipulators are prone to fatigue. Therefore, modelling deflections in the mechanical structure under different operating conditions can help in optimizing the manipulator geometry and in selecting the right material for the same.

CAE used in many fields, for example, car, flying, space, and ship building ventures and bio-mechanical enterprises. It includes use of wide scope of software's to play out the errands. The market is overflowed with different software's which are based on the procedures, for example, finite element method (FEM), computational fluid dynamics (CFD), multi-body dynamics (MBD) and optimisation.[3]

OBJECTIVES

The objective of the paper is given as follows-

- To carry out static and modal analysis of IRB 140 manipulator for four different materials and justify the best of them.
- To carry out dynamic analysis of IRB 140 manipulator for maximum acceleration over a specific span of time.
- To carry out the static analysis of gear assembly for four different materials and justify the best of them.

Finite element method is one class of methods where this contribution from each node is obtained via constructing small finite elements with these nodes as vertices. These are done by connecting the nodes and constructing elements. The obtained structure is called a mesh as it resembles a mesh. And then based on the geometry each element contributes to some percentage of the total displacement of that node. This is achieved by associating equations called shape functions to each node in each element.Finite element analysis of any strong segment includes pre-processing, solution and post processing stages.[3,5]

GEOMETRY

A 3-D model of a six-axis mechanical robot was used to do CAE. The robot model ABB IRB 140 used in this paper is a compact, powerful industrial robot that can handle a variety of applications such as arc welding, spraying, material handling, cutting/debarring, die casting etc. It is a 6 rotational axis robot with a payload of 5 kg and multiple mounting options. The axis 5 reach of the IRB 140 is long at 810 mm. Figure 2 shows 3-D CAD model of an industrial robot ABB IRB 140. Also, the IRB 140 represents the configuration of most widely used six-axis industrial robots. The IRB 140 has good flexibility (with respect to joint limits) and a large work envelope which is useful in solving the functional work space problem. [8]

The gear assembly of ABB IRB 140 type 3-D model was designed in Solidworks.15. Initially, the assigned materials of all the main bodies of the manipulator joints and links are cast iron. The material used for the gear assembly is stainless steel. The 3-D models of the robot and gear assembly are shown in figure 2 and figure 3.

The same model of ABB IRB 140 was used to do CAE for different alternating materials like polyethylene, Epoxy granite and stainless steel. For the gear assembly, different materials like nylon, aluminum alloy and copper were used. The same are shown in table 1 with respective properties.

Table 1: Materials used for Analysis

Manipulator Material	Gear Material
Cast Iron Density-7200 kg/m^3 Yield strength-110 GPa	Nylon Density-1140 kg/m^3 Yield strength-3.01 GPa
Polyethylene Density-950 kg/m^3 Yield strength-1.1 GPa	Aluminum alloy Density-2770 kg/m^3 Yield strength-71 GPa
Epoxy granite Density-1530 kg/m^3 Yield strength- 4.075 GPa	Stainless steel Density-7750 kg/m^3 Yield strength-193 GPa
Stainless steel Density-7750 kg/m^3 Yield strength-193 GPa	Copper Density-8933 kg/m^3 Yield strength- 128 GPa

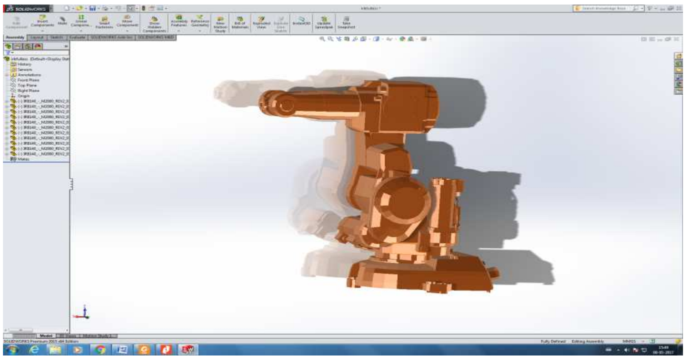


Figure 2

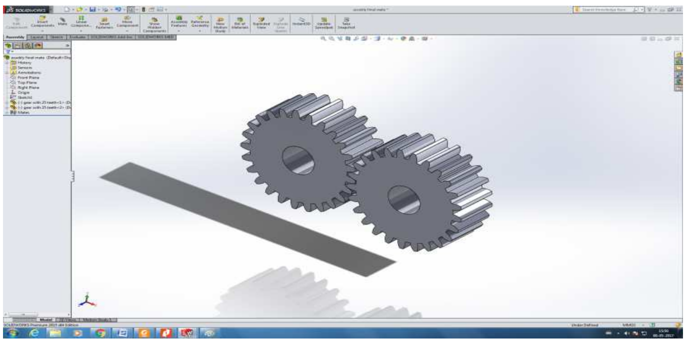


Figure 3

FINITE ELEMENT ANALYSIS

After the geometry is finalized the next step is to do the pre-processing. Here the mass properties of the geometry are utilized as data sources. In this stage the manipulator was broken into smaller elements and nodes. This process is known as meshing. After the manipulator has been meshed properly then it was subjected to the boundary conditions and loads. This stage is called solution. After the boundary conditions and loads had applied and the results were obtained it is necessary to interpret the results. This was done under the phase of post-processing. It is critical to decide how stiff the structure is. The geometry of the model is utilized for this type of decision.

Static Structural Analysis

So as to consider the static basic conduct of the robot on the modular static analysis an underlying static investigation examination was finished. A most extreme payload of 5 kg (49.033 N) set on the mounting spine of the end-effectors was considered. The material used for the whole manipulator is cast iron.

Model Pre-Processing

The CAD model of industrial robot along with its gear assembly will be imported in the ANSYS software. Before importing, the model will be saved in STEP (.stp) format, IGES (.igs) format or Para solid neutral data exchange translator. After this, pre-processing will be done and the model will be meshed. Then boundary conditions and payloads will be applied to find out the results.

Meshing

The model was meshed using dominant hexahedral components with 8 nodes. The element size 5 mm was utilized (Figure 4).In meshing, the numbers of nodes are 2054524 and numbers of elements are 597652. Figure 5 demonstrates the loads considered in the underlying static examination. The total calculated deformation was beneath 0.018 mm and the most maximum equal stress was 1.859MPa in the case of cast iron, as shown in figure 6.1and 6.2.The maximum equivalent strain is 1.739 X 10^{-5}.

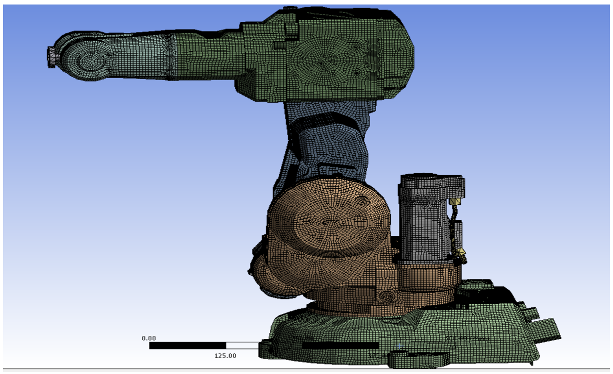


Figure 4

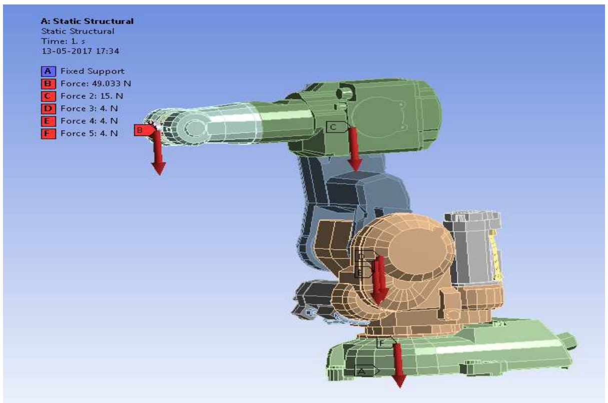


Figure 5

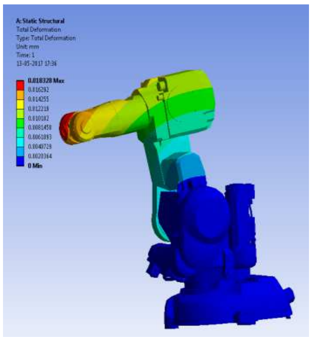


Figure 6.1

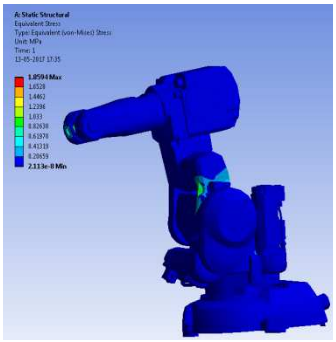


Figure 6.2

For the same pre-processing and solution of the robot, the material used was stainless steel. After analysis, the maximum deformation is 0.01048 mm, the maximum equal stress was 1.841MPa and the maximum equivalent strain is $9.8127X10^{-6}$. For Epoxy Granite the maximum deformation is 0.025 mm, the maximum equal stress was 1.847MPa and the maximum equivalent strain is $2.37 \text{ X } 10^{-5}$ and for polyethylene the maximum deformation is 1.851 mm, the maximum equal stress was 1.77MPa and the maximum equivalent strain is 0.0016.

Neither the maximum stresses, nor the displacements have high values that could influence the robot accuracy and precision for this preparatory calculation.

Modal Analysis

The modal examination decides the vibration attributes, for example, normal frequencies, mode shapes of the structure and support elements. It is additionally the beginning stage for any another dynamic study. The normal frequencies and the mode shapes are essential parameters in the plan of a robot with respect to the dynamic loading conditions. At the point when the modular investigation utilizes the consequences of a static examination as data sources, the modular investigation is named modular pre-stressed. This is the approach utilized as a part of the present review, where the initial five natural frequencies (it tends to vibrate when disturbed) where registered (Figure 7). [2, 5]

The initial two modes (f_1 = 21.36 Hz and f_2 = 43.92 Hz) are twisting methods of the end-effectors-arm-interface arm get together in the horizontal plane, and in the vertical plane separately, brought on by the wrist. The following two modes are twisting vibrational methods of a similar get together created by the arm (f_3 = 58.43 Hz and f_4 = 186.66 Hz), while the fifth mode (f_5 = 204.95), is complex twisting modes including likewise the turning section, brought on by the vibration of the arm and the connection arm, separately. Last mode has high natural frequency, so its impact on the robot

positioning accuracy is quite low.

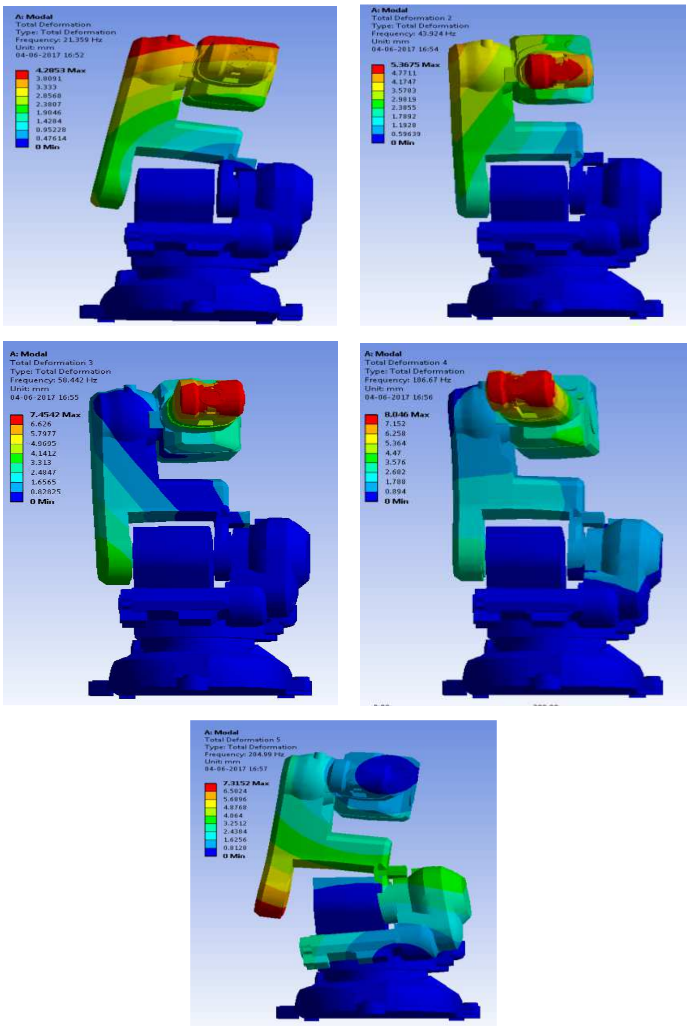


Figure 7

The five modes of natural frequencies for epoxy granite material used in manipulator are f_1 = 8.9 Hz, f_2 = 18.324 Hz, f_3 = 24.342 Hz, f_4 = 77.821 Hz and f_5 = 85.825 Hz.

The five modes of natural frequencies for stainless steel material used in manipulator are f_1 = 27.41 Hz, f_2 = 56.01 Hz, f_3 = 74.34 Hz, f_4 = 237.78 Hz and f_5 = 262.82 Hz.

The five modes of natural frequencies for polyethylene material used in manipulator are f_1 = 6.06 Hz, f_2 = 12.07 Hz, f_3 = 15.885 Hz, f_4 = 50.932 Hz and f_5 = 57.918 Hz.

From the above natural frequencies for different materials assign for manipulator, the last frequency at fifth mode had high value. Therefore its influence on the manipulator positioning accuracy is low.

Rigid Body Dynamics Analysis

The kinematic analysis is a key review on the conduct of mechanical frameworks. To find the end-effectors position of a robot, the analysis of halfway joint positions must be performed. The kinematic examination likewise decides the joint loading and checks the robot displacements, velocities and acceleration in the work space. This type of analysis

was conducted in Rigid Dynamic module. This module solver automatically creates its local reference system in each of the part of manipulator. Each joint is attached to another joint with a proper reference system.

The kinematic analysis was performed for different positions of robot within the certain time constraints. The robot moves in the work envelope for 25 s and the maximum payload was 5 kg. The equations were solved using Runge-Kuta 4 integration method. The various positions of revolute joints of the robot are shown in figure 8. The joint movement shows its movement relative to reference and mobile part. For the time span of 25 s the positions of robot at different rotations are shown in figure 9.2 and 9.3. The total acceleration along with the values is shown in figure 9.1 and the respective graph for the same is shown in graph 1 for 25 s. In this analysis, the maximum acceleration peak at 22.72 s.

The total displacement and total velocity of the robot with respect to its home position (from end-effector) shows in figure 9.4 and figure 9.5 and the graphs of total displacement and total velocity of the robot from its initial position is shown in graph 2 and graph 3 for the time span of 25 s.

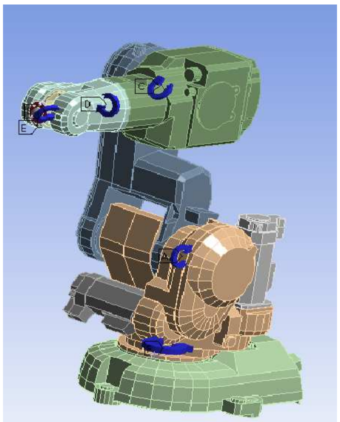


Figure 8

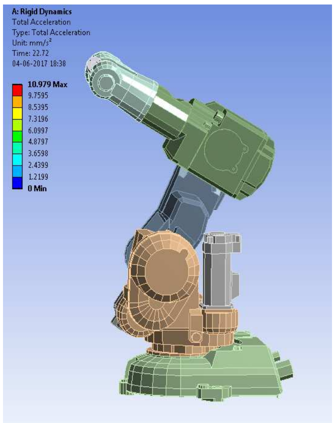


Figure 9.1

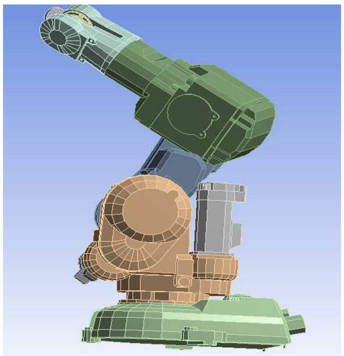


Figure 9.2

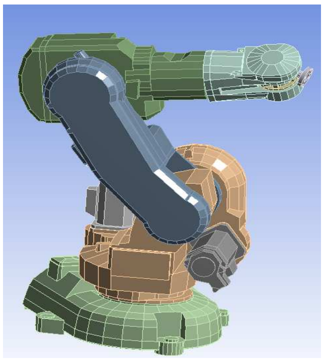
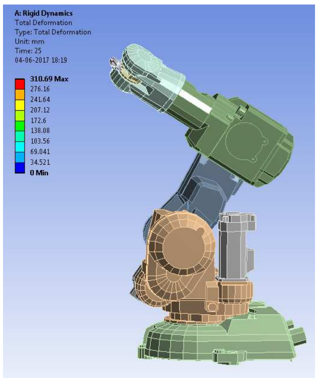


Figure 9.3



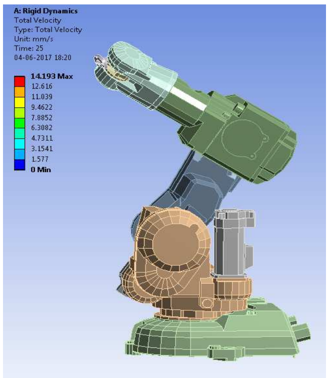
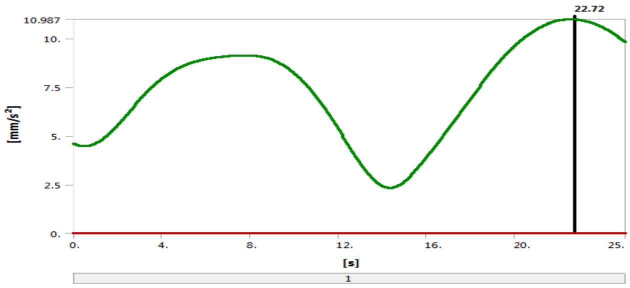
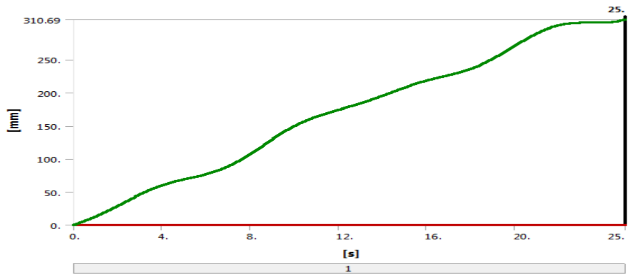


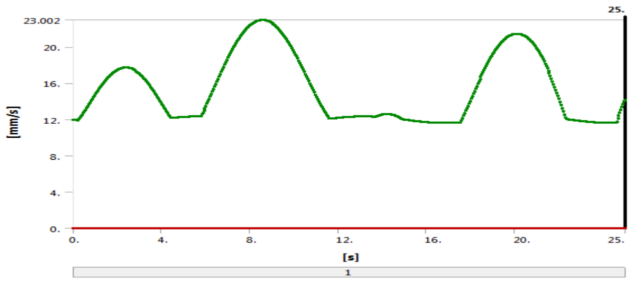
Figure 9.4 **Figure 9.5**



Graph 1: Total Acceleration

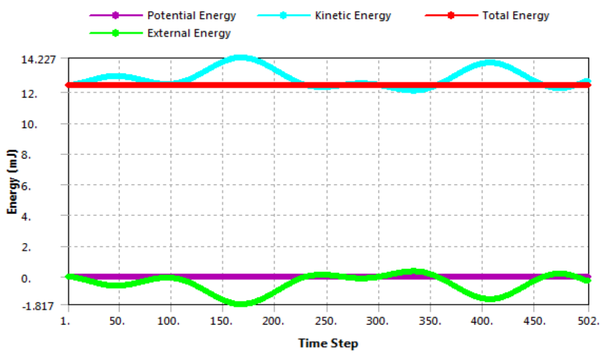


Graph 2: Total Displacement

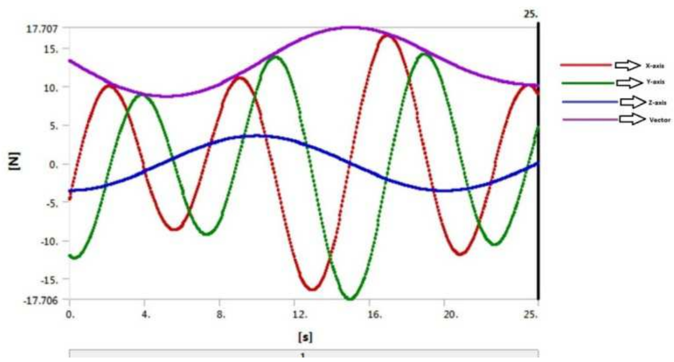


Graph 3: Total Velocity

The energy probe shown in graph 4 proved that the potential, kinetic and external energy at all the integrations points were well balanced for the given span of time. Moreover the analysis was an efficient approach to understand the overall dynamic behavior and generate boundary conditions for structural analysis. The resultant reaction force for this analysis was found to be 17.707 N as shown in graph 4. It was found that by apply 17.707 N forces on end effectors in static structural analysis the values of von mises and total deformation were 0.68 MPa and 0.008 mm respectively.



Graph 3: Energy Summery



Graph 4: Reaction Forces of End Effectors

GEAR ASSEMBLY OF INDUSTRIAL ROBOT

The Industrial robot is a combination of drive system, sensors, power supplies & power storage system, software and feedback control system. The most important part of robot control is transmission system. The transmission system used in robot to transmit power and motion consists of gears, chains, timing belts, metal belts, cables and pulleys and linkages. The most important part of transmission is gear. Gears are a toothed machine part, such as a wheel or cylinder that meshes with another toothed part to transmit motion or to change speed or direction. Interference in gear occurs when the addendum of pinion enters the non-involutes dedendum of mating gear. Spur gear is usually used in the industrial robot for torque transmission. There are a few sorts of stresses present in stacked and pivoting gear teeth. We need to consider every one of the potential outcomes, so that the gears are corresponding to keep every one of the worries with in design limit. The finite element method is accurately providing the stress analysis results.

Analysis of Spur Gear

The set of 3-D model of spur gear is generated using Solid works CAD software. The CAD model of gear assembly will be imported in the ANSYS software. Before importing, the model will be saved in IGES (.igs) format. After words, pre-processing will be done and the model will be meshed. The material assign for spur gear assembly is stainless steel. The meshing of gear assembly is shown in figure 10. The table 2 shows the dimensions of spur gear.

Table 2: Dimensions of Spur Gear

Parameters	Measurement Unit	Symbol	Input Values (For Both Gears in Assembly)
No. of Teeth	unit less	Z	25
Pitch Circle Diameter	mm	D	50
Pressure Angle	degree	ϕ	20
Addendum Diameter	mm	RA	54
Dedendum Diameter	mm	RD	47.50

Boundary Condition

Friction-less support is connected on internal edge of the left gear. Friction-less support is connected on the internal edge of right gear to permit its tangential rotation yet confine from radial translation. Moment of 25 N-m is connected on the internal edge of right gear in clockwise direction as a driving torque. The value 25 N-m is the operational torque value. [8]

The result obtained from ANSYS shows the equivalent stress which gives the maximum value of 145.32 MPa as shown in the figure 11. The value of the maximum equivalent strain is 8.23 X 10^{-4} as shown in figure 12. The total deformation calculated is 0.020mm as shown in figure in 13.

For the same pre-processing and solution of the gear assembly, the material used was nylon. After analysis, the maximum deformation is 1.99 mm, the maximum equal stress was 60.175 MPa and the maximum equivalent strain is 0.0217. For Aluminum alloy the maximum deformation is 0.054 mm, the maximum equal stress was 145.09 MPa and the maximum equivalent strain is 2.23 X 10^{-3} and for copper the maximum deformation is 0.037 mm, the maximum equal stress was 146.15 MPa and the maximum equivalent strain is 1.3 X 10^{-3}.

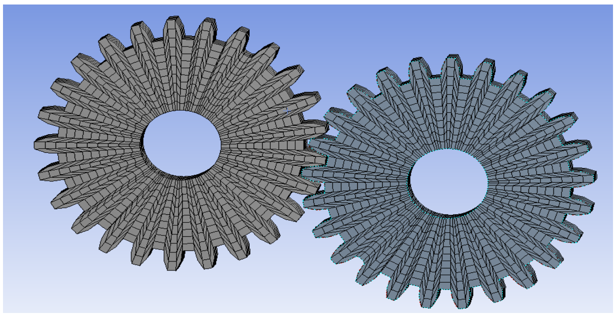


Figure 10: Meshing of Spur Gear Assembly

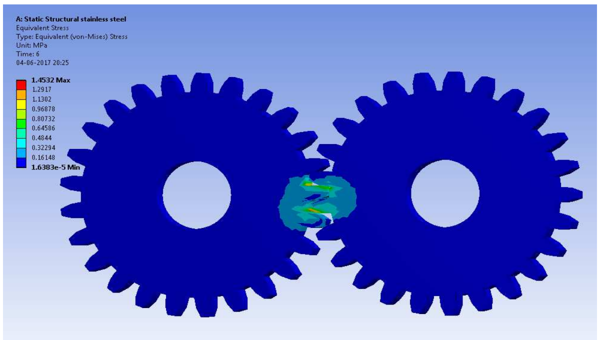


Figure 11: Maximum Equivalent Stress

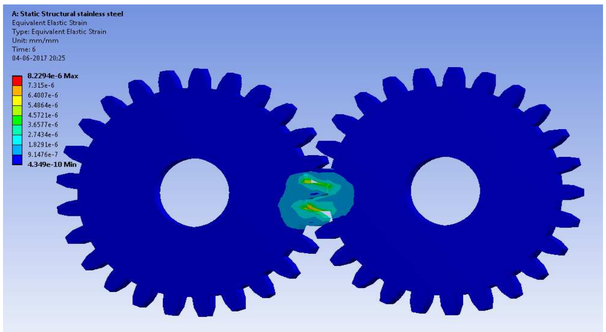


Figure 12: Maximum Equivalent Strain

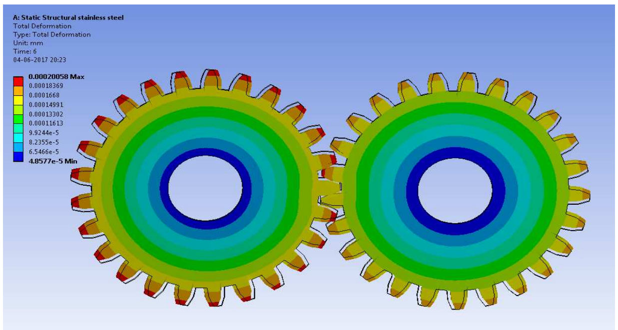


Figure 13: Total Deformation with Respect to Initial Model

The black color sketch shows the initial position of the gear assembly. The results values are very less so there is no such effect of on gear assembly when subjected to250Nm torque. The red color shows the maximum value and blue color shows the minimum value of the equivalent stress as shown in Figure 14.

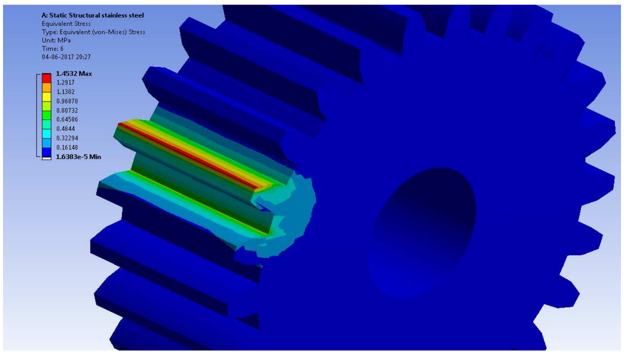


Figure 14

CONCLUSIONS

The CAD, CAE study was performed on a 6-DOF model of an industrial robot and its transmission assembly system. FEA investigation is completed utilizing ANSYS software to assess the static and dynamic performances of the manipulator, which approve the precision of the consistence, variability of joints and kinematics models also.

In static, modal and dynamic analysis of an industrial robot four materials viz. cast iron, polyethylene, Epoxy granite and stainless steelwere used in the simulations. Out of four the best material to do operations was cast iron because of its hardness and damping capacity characteristics.

In the case of gear assembly used in manipulator, four materials viz. stainless steel, nylon, aluminum alloy and copper were applied in the static analysis. Out of the four the best suited material for gear assembly in industrial robot was stainless steel because of its strength, durability and corrosion resistance properties.

Computer aided design helps to reduce the cost of production and completion time for the same. Meanwhile they help to ensure that the things turned out with higher quality and better strength. Design with CAE will likewise help outline and building groups oversee execution suggestions and fractures of their design. Also, designers can make utilization of computer based simulations to refine and assess their physical model. CAE help makers identify flaws with the item on time, so issues can get arrangements immediately.

REFERENCES

1. *M. Akdağ,* Design and analysis of robot manipulators by integrated CAE procedures*, DEÜ Fen Bilimleri Enstitüsü, Doctoral Thesis, 2008.*
2. *Cristina Pupaza, George Constantin and ŞtefanNelgrila, Computer Aided engineering of Industrial Robotos, Proceedings in Manufacturing Systems, Vol. 9, Issue 2, 2014 pp. 87-92.*
3. *B. Reddy, R.G. Brioso,* Automated and Generic Finite Element Analysis for Industrial Robot Design*, Thesis, Linköping University, 2011.*
4. *M. Zglinska, G. Orzechowski and J. Fraczek,* Dynamical analysis of the deformable power transmission mechanism using mixed FEM and MBS method*, Journal of KONES Powertrain and Transport Vol.15, No.1 2008, pp.235–245.,*
5. *ANSYS Inc.,* Design Exploration User's Guide*, Release 15.0, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, USA, February 2014.*
6. *T. Shoba Rani, T. Dada Khalandar (2013), Spur Gear, International Journal of Computational Engineering Research, Volume 03, Issue 11, pp. 7-12.*
7. *Bharat Gupta, Abhishek Choubey and Gautam V. Varde (2012), Contact Stress Analysis of Spur Gear, International Journal of Engineering Research and Technology, Volume 1, Issue 4, pp. 1-7.*
8. *ABB Products.ABBIRB140Robots(Robotics). www.abb.com/product/seitp327/b0168cc2e1ea9fd5c125717900223403.aspx*
9. *McGuinn, Jack, Seniour Editor, these Bots are Cutting –Edge, Power Transmission Engineering, October 2008, pp.32-36.*
10. *S. Niku, Scheme for Active positional correction of robot Arms, Proceedings of the 5th international Conference on CAD/CAM, robotics and factories of future, Springer verlag, pp. 590-593, 1991*
11. *Koren, Yoram, Robotics for Engineers, Mcgraw-hill, 1985.*
12. *S.B Niku, Introduction to Robotics: Analysis, Control, Application, Wiley, 2014*